



CNC Routing Overview (9 March 2019)
J.Sikorski (john2pt0)

Objectives	2
Safety	2
Computer Numeric Controlled (CNC) Machining	3
Taking a project from design to machine ready	6
Feeds & Speeds in the Woodshop	6
CAD / CAM (short intro. Much more info is on the Vectric website.)	9
Postprocessing	17

Objectives

This document will introduce you to:

- a) the **safety precautions** for using the CNC router
- b) the **terminology** of CNC machining (**CAD, CAM, postprocessing**)
- c) **taking a project from design to machine ready** (GWizard, VCarve)
- d) the **setup and running** of a project (**Mach 3**)

This is not a substitute for taking the CNC course.

You will need to take the course to get our makerspace VCarve registration number to use VCarve with our CNC router.

To be **certified** to use the Hive 1 3 CNC router, you will need to take the Hive 1 3 CNC Intro Course and then arrange for a Hive 1 3 **CNC machinist** to observe you running a project. Post your proposed time & date on the hive13.org website mailing list page to ask for coverage.

Safety

Protect yourself:

safety glasses, ear protection,
no hair, lanyards or hoodie ties dangling out it front
tie up and tuck in the back of your shirt
no gloves!
closed-toe shoes

Protect your colleagues:

Secure the mill (cutter).
Secure the material.
Avoid loose wood with loose knots or cracks.

Protect the machine:

Secure the mill.
Secure the material:
Use nails at least 1/2" longer than the thickness of your material.
Nail every 6-12", including along cuts along the center of the board.
If using mills > 3/8" in diameter, consider using screws as hold downs.

Don't cut more than 0.04" (1mm) into the spoil board

(0.02" should suffice for MDF, plywood, plastic and aluminum)

Do Not Leave the Machine While It's Running!

Computer Numeric Controlled (CNC) Machining

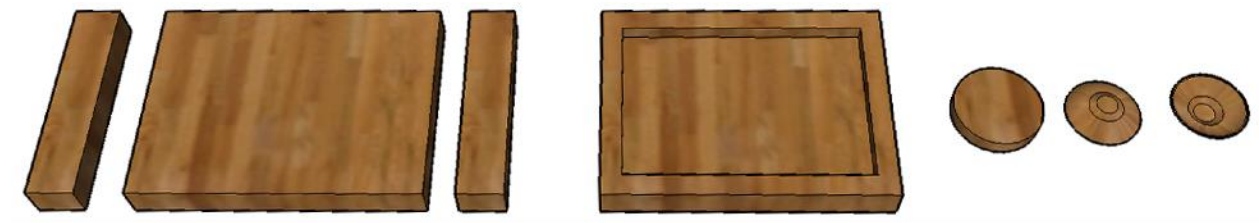
CNC machining is “subtractive” (like wood carving) and can cut 2D, 2.5D, and 3D tool paths.

2D - cuts in one plane (like cutting a board to size on the table saw or cutting holes with the jig saw)

2.5D - moves the mill to different levels and then does 2D cutting at that level to form holes or pockets.

(like using a hand held router to cut a serving tray from a single board; you level the entire top of the 12” x 8” board to set the thickness, then level the middle area of the board 1/2” lower, leaving edges elevated 1/2” along each side.)

3D - cutting objects smoothly rounded on one or both sides (like shaping a saucer, as one can on the lathe but here you lay the wood flat to smooth one side, then flip it over to smooth the other)



2 D

2.5D

3D

CNC work include 3 stages:

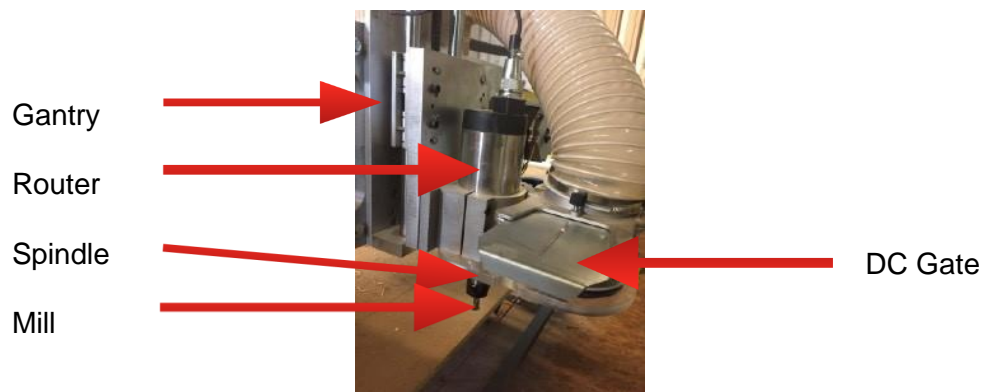
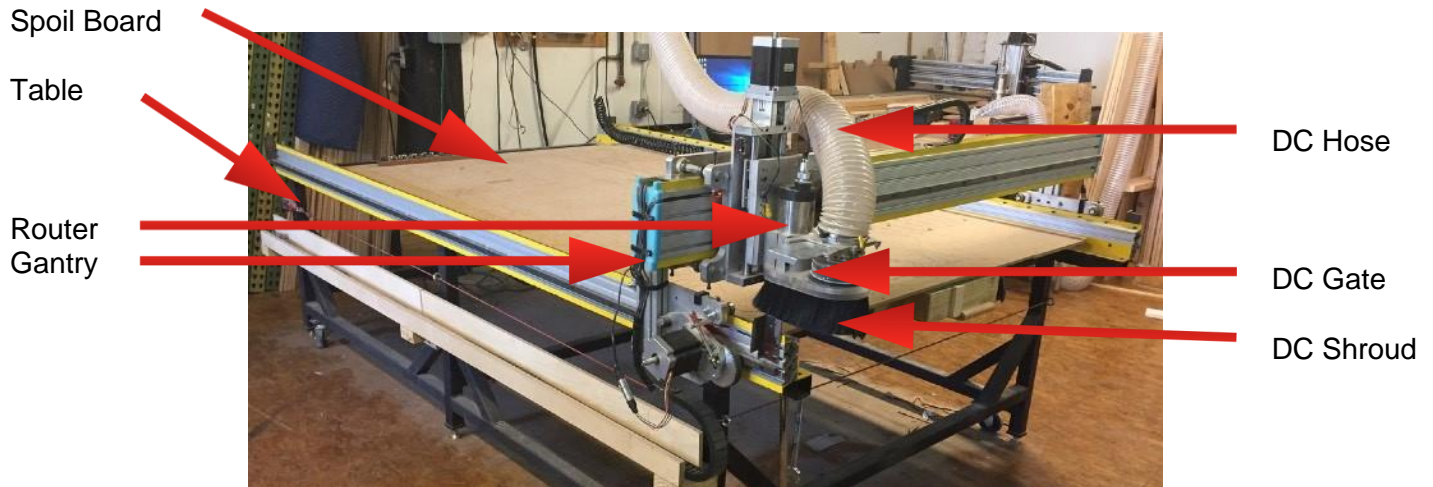
CAD (computer aided design) - draw up the part in the software of your choice

VCarve, Illustrator, Sketchup, Fusion 360, Inkscape, etc.

CAM (computer aided manufacturing) - decide what mills will cut which features, that is, define the toolpaths in software that provides this function **VCarve**, Fusion 360, etc.

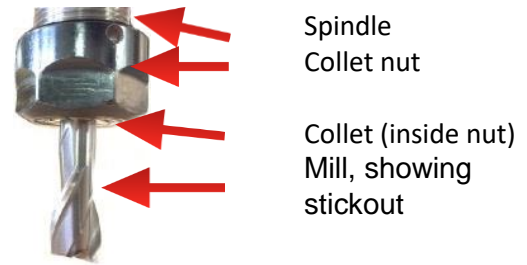
postprocessing (generate the text file that drives the CNC machine) run a postprocessor to produce the text file (GCode) that will move the spindle to cut the toolpaths with the mills, feeds and speeds you specified in CAM **VCarve**, Fusion 360, etc.

The Hive13-built CNC router has standard parts:
the **table** which supports the **spoil board**,
the **gantry** which moves the **router** in 3 axes in response to the **GCode** text file,
the **spindle**, which holds the cutting **mill** in the collet (compressible washer) and spins the mill,
the **dust collector** hose, **gate** and **shroud** (to...collect dust ;^) the emergency stop cord (the purpose of the **E-cord** is left as an exercise for the reader...).

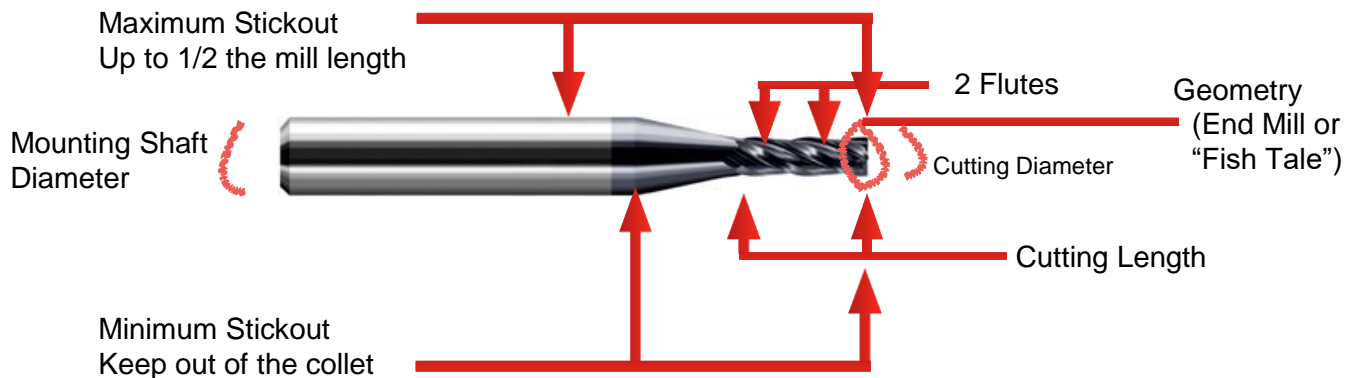


The CNC router works by moving the mill at a **feed** rate (GCode “**F**”)while spinning it at a **speed** rate(“**S**”), both of which are specified in the GCode text file of the **toolpath** along which it is moving. (VCarve GCode lines N150 & N190 specify the speed and the feed, respectively).

When not cutting, the mill is moved quickly between tool paths at an automatically set speed called the “**rapid**”. **It is important that your toolpath raises the mill be above all obstacles** (nail, screws, clamps) **when performing a rapid**.



Mills (similar to “router bits” in the woodshop) are specified by: **diameter** (of cutter and also mounting shaft, if different), numbers of **flutes** (cutting edges, 1 or 2 for a CNC router), **geometry** of the end of the cutter (end mill, ball nose, etc), **cutting length** (length of the flutes), **stickout** (the length sticking out beyond the collet - should not exceed 1/2 the total mill length) and **material**: high speed steel (HSS) or carbide (you don’t need “coatings” for wood or plastic).



A starter set of mills could contain a couple of 3/16-3/8” end mills & a 60° v-bit with the same size shaft (so you won’t need to change collets when you change mills).

We **recommend against** using hand-held (or router-table) **router bits** in the CNC router. They are not built to tolerate the lateral stress of CNC machining and can break and send metal and cutters flying!

The only exception is hand-router **V-bits**. Although V-style **mills** are preferred, V-bits are considered safe in a CNC router when used for engraving lettering, as long as only a small part of the bit ($\leq 0.125''$ (3mm)) **cuts into the wood to engrave** the letters, minimizing lateral stresses. (i.e., the **Depth of Cut** is $\leq 0.125''$)

Mills are not drills. Mills are made to cut horizontally. **Don’t try to drill holes** with them. (Cutting out a **circle** that has a **larger diameter** than the mill is OK, it’s still a horizontal cut.) If you want to drill a hole, put a **drill bit** into the collet & use a “**peck drilling**” toolpath.

Taking a project from design to machine ready

As noted above, to calculate a toolpath, you need to have a feed, speed, and possibly a plunge rate. (The plunge rate is **not** for drilling, it determines how fast the mill can change depth while moving laterally.) When calculating Feeds & Speeds, you will need the **cutter** diameter, **number of flutes**, **stickout** and **material**.

Stickout was mentioned above as a range of values, i.e., “at least the cutting length, but no more than half the length of the mill.” The less the stickout, the less the vibration of the mill, and the cleaner the cut. It was also noted that you need the stickout to calculate the feed and speed. You need the feed and speed before you can run the mill on a toolpath, but you won’t know the actual stickout until you install the mill in the collet in the spindle. What?!

The answer is that, on a CNC router, you just need to get within about 1/8” of the stickout you used in your calculation when installing the mill. Install it and get as close to there top of the flutes as you can, without worrying too much. If you’re too far off, the mill may break, or the finish may be rougher than you expected. As with much of CNC routing, make an adjustment and try again.

Feeds & Speeds in the Woodshop



GWizard

(Note: GWizard is often updated and may be slightly different than described below. Check cnccookbook.com for info.)



Have the following info available:

material you’re cutting

tool: material (HSS or carbide), cutter diameter, # of flutes, stickout

cut: depth & width(stepover)

Note: GWizard can be set to SAE or metric via the Setup tab - check that it matches your units!

Open GWizard; click on the Feeds/Speeds tab.



Machine From the drop-down menus, choose:
Generic CNC Router
 Material: **Wood** (hard, soft, MDF, plywood), Plastic or Aluminium, wax
 (no other materials on the this machine!)
 Tool: **Carbide** Endmill or **HSS** Endmill

The screenshot shows the 'Machine' and 'Tool' configuration panels. The 'Machine' panel has a dropdown menu set to 'Generic CNC Router' and a 'Material' dropdown set to 'Wood: Plywood'. The 'Tool' panel has a 'Tool' dropdown set to 'Carbide Endmill', a 'Tool Dia.' input field with '0.25', a 'Flutes' input field with '2', and a 'Stickout' input field with '1'. Red arrows point to each of these five elements.

Tool Enter:
Tool Dia. (dia. of cutting end) (Check default units!)
Flutes: 1 for plastic or aluminium; 2 for wood and wood products
Stickout: (See above comments.) (Check default units!)

Mfg Skip for basic setup
 Mini Calc Skip for basic setup

Cut Enter:
 Cut Depth: wood: start with the radius of the cutting end (“Axial Engage” 50%)
 plastic or aluminium start with **1/5** the **radius** and see how it goes
 Cut Width: less than the radius of the cutting end (“Radial Engage” 12.5% - 48%)
 (“Cut Width” is often called “stepover”. It is the width of the cuts which occur after the first cut.)
 (Verify or enter the following value to match the default setup for our CNC router.)

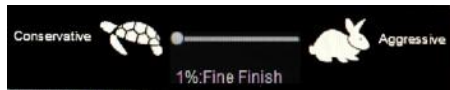
Limits
 RPM Limit: **20000**
 Min RPM: **0**
 Feed Limit: **200** (Skip the rest for basic setup)

The screenshot shows the 'Cut' and 'Limits' configuration panels. The 'Cut' panel has a 'Cut Depth' input field with '0.125' and a 'Cut Width' input field with '0.12'. The 'Limits' panel has several input fields: 'HP Limit' (2), 'RPM Limit' (20000), 'Min RPM' (3000), and 'Feed Limit' (200). Red arrows point to the 'Cut Depth' and 'Cut Width' inputs, and three red arrows point to the 'RPM Limit', 'Min RPM', and 'Feed Limit' inputs.

HSM Skip for basic setup

Tips

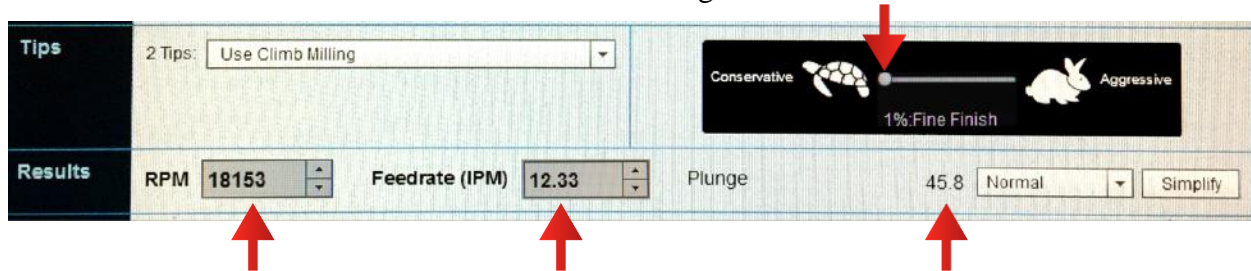
May have some interesting suggestions



(Tortoise - Hare panel)

(Click on the slider & adjust with the mouse or arrow keys between 1% and 100%.)

Adjust the % until one or both numbers (RPM and Feedrate) turn red, then back off until both are black again.



This will show you the calculated max and min RPM (“Speed”) & Feedrate (“Feed”)(in Inches per Minute, IPM).

It may also show a Plunge rate.

The results at 29% are estimated by the software as giving the optimal compromise for maximum Feedrate still resulting in a smooth finish. This is just a guideline. YMMV.

If the RPM (“Speed”) stays at 20,000 (in the red) even at the lowest Feedrate, adjust the panel % and find the max & min Feedrate that appear in black. Use these as your limits at 20,000 rpm.

You will use this data in the “**Edit**” panel after you “**Select**” a tool (and click **OK**) in a VCarve toolpath. (see below)

The higher the Feed, the faster, but rougher the cut.

Play with the Feed (within the limits you found in GWizard) and see what gives satisfactory results on the material you’re using.

Results

(Write these down for use in VCarve.)


RPM (“Speed”)

Feedrate (IPM) (“Feed”)

Plunge

(You will also need the cutter diameter and stickout you used above for use in VCarve.)

Cut KB

skip for basic setup 

CAD / CAM (short intro. Much more info is on the Vectric website.)

Launch VCarve Pro 9

(Download the VCarve Pro trial version and enter your makerspace registration.)

If you are just postprocessing the toolpaths in your pre-saved project file (.crv) to generate the GCode, go to **Postprocessing**, below.

If you are resuming a prior project, click **Open an existing file**.

If you are starting a new project, including importing a design from, say, Inkscape, click **Create a new file** to fill out the **Job Setup** panel.

Create a new file - this will open the **Job Setup** panel. Check **Single Sided**
(You can do 3D machining on one side, or on two sides.
See tutorials on the Vectric website.)

Enter **Width** (x) and **Height** (y). and **Thickness** (z) of material.
(VCarve will calculate fractions for you: e.g. enter "18(space)1/16=" the result is 18.0625)

Verify your **Units**.

Set **Z Zero** to **Machine Bed**.

Set the **XY Datum Position** to the lower left corner of your material. (It helps to mark this on your material.)

Click **OK** - this will open the **Drawing** panel.

If you need to get back to the **Job Setup** panel, click on the white square icon in the second row under **File Operations**.

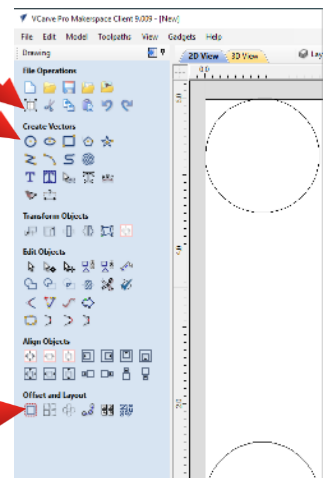
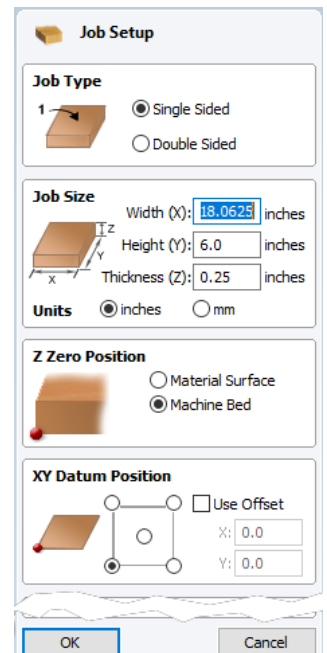
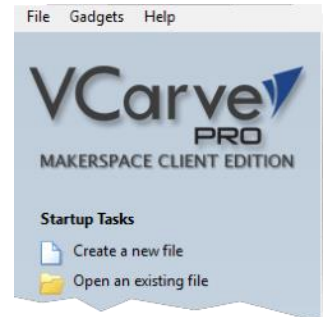
CAD (Computer Aided Design)

Mark the placement of the hold downs (nails, etc.) with 1.5" circles using the circle icon under **Create Vectors**.

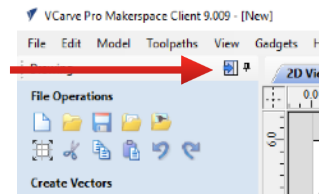
Import (under **File**, in the menu bar) or create the vectors that will define your design.

If you import vectors, check for redundant or open vectors.
Eliminate the former and close the latter. See the Vectric website for vector editing info.

Use **Offset** under **Offset and Layout** to make a path for cutting your project out of the material, if needed. Offset it by the radius of the mill you will use.

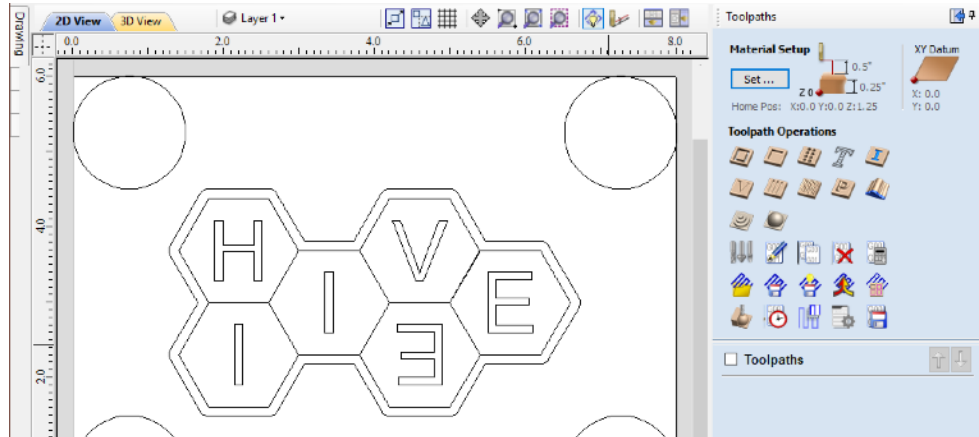


Click the right-pointing blue arrow icon (**Switch to Toolpath commands**) at the top of the **Drawing** panel to get to the **Toolpaths** panel.



CAM (Computer Aided Manufacturing (toolpaths))

Before creating toolpaths, we need to set up our parameters for the GCode.



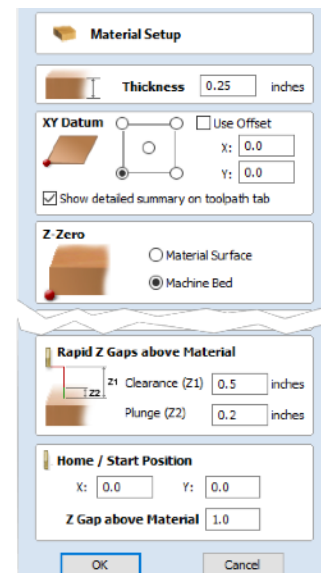
Under **Material Setup** in the **Toolpaths** panel, click **Set...**
This will open the **Material Setup** panel.

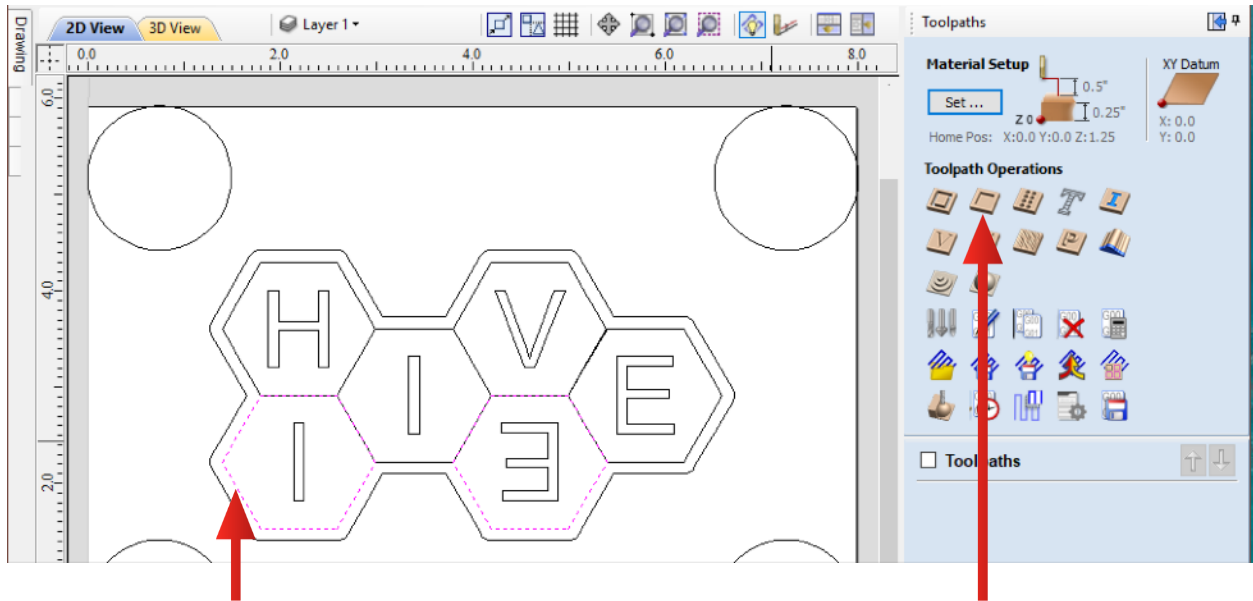
Check that the **Thickness**, **XY Datum** corner, and **Z-Zero** location match your settings in the **Job Setup** tab.

In **Rapid Z Gaps above Material**, enter values for
Clearance(Z1) and **Plunge(Z2)**
Z1 to clear nails & screws, usu 0.5-1" (or above any clamps!)
Z2 to prevent the descending mill from crashing into material, 0.2"

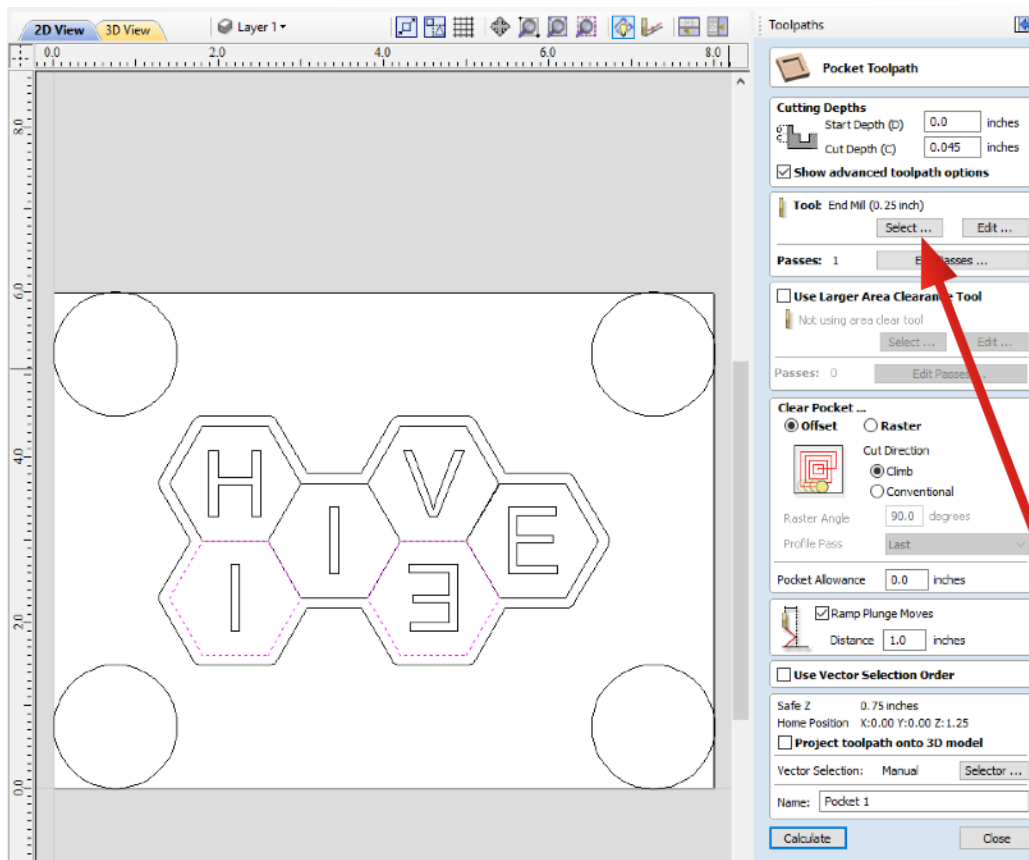
In **Home/Start Position**, enter values for
Z Gap above Material
Calculate as **Z Gap above Material** as **Thickness + Z1 + Z2**
(We rounded up from 0.95" to 1.0")

Click **OK**. This will take you back to the **Toolpaths** panel.





Choose each vector in your design in turn and click on a toolpath icon (here a **Pocket** toolpath).



Define the **Start depth**.
(0 = the top of your material.)

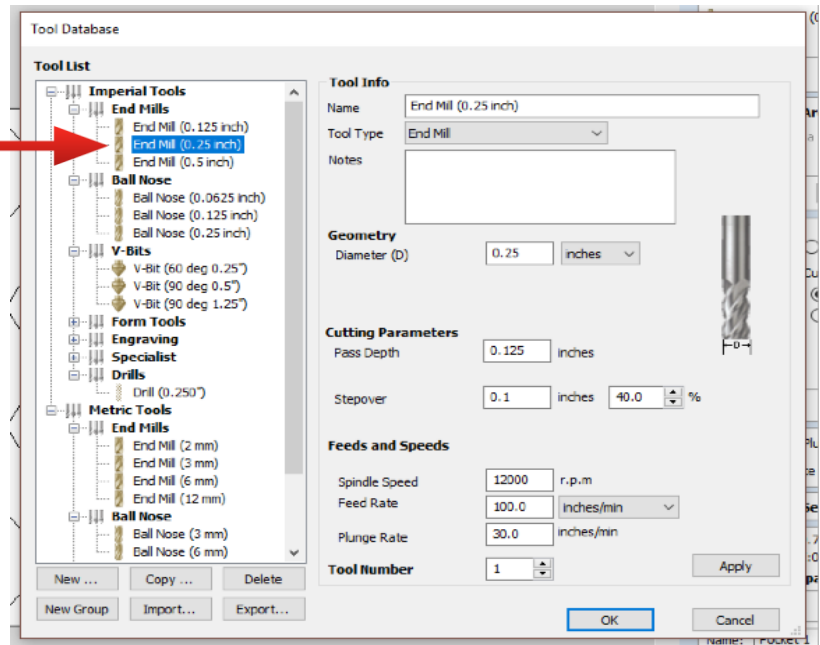
Here we set a depth for the pocket we are cutting, of 0.045"

Choose a tool type from the crib via **Tool: Select**

Choose a tool. Then Click OKSpecify the parameters for *your* mill via **Tool: Edit...** (next to **Tool: Select**, above)

Choose a Tool.

Click **OK**.



Specify the parameters for *your* mill via
Tool: Edit... (next to **Tool: Select**, above)

Verify that **Diameter** matches the cutting diameter of your tool.

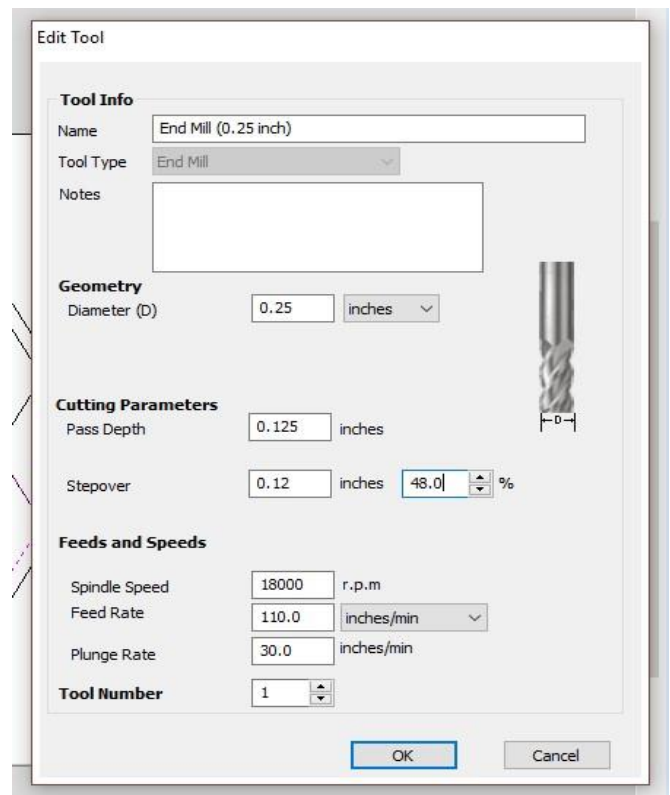
Start (in wood or plastic) with a **Pass Depth** that is no greater than the cutter radius.

(1/5 of the radius for aluminium.)

Stepover should be less than the **Pass Depth**

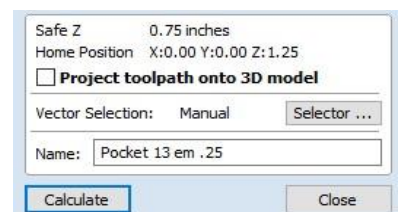
Enter the **Speed**, **Feed** and (if applicable) **Plunge** for your tool. Be sure the units for **Feed Rate** are correct (**inches/min**).

Click **OK**

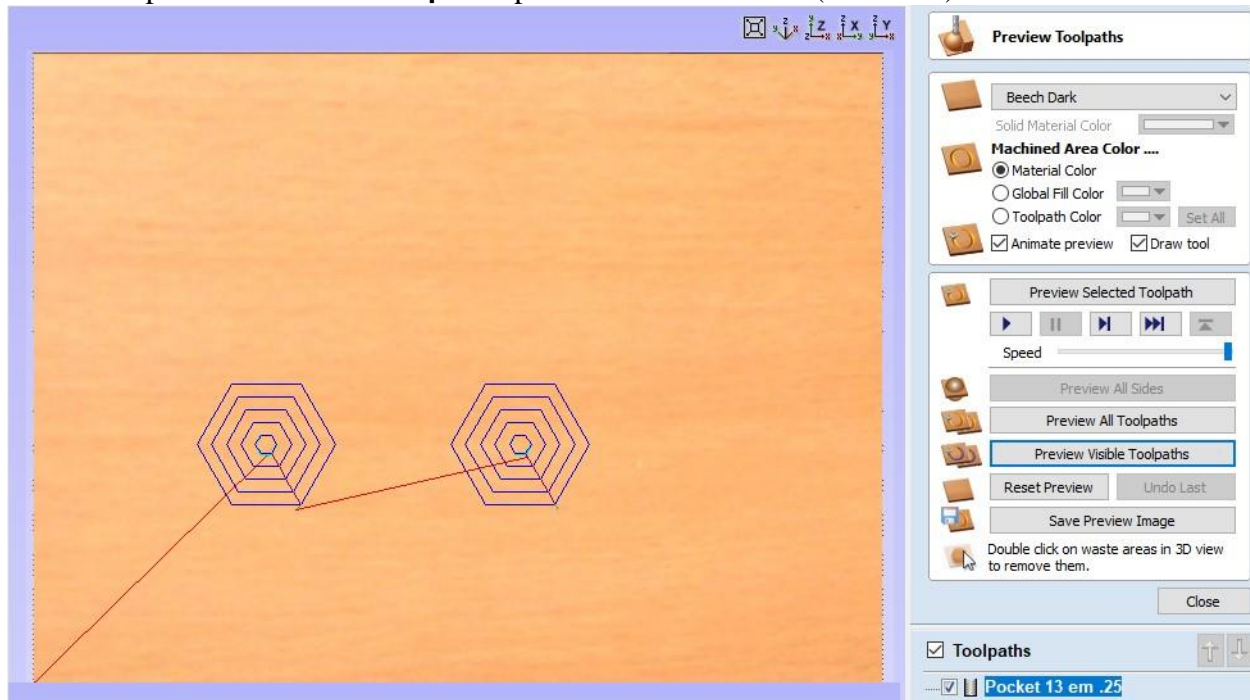


Name the toolpath so you know what project, what vector and what mill (type & size) are used.

Click **Calculate**.



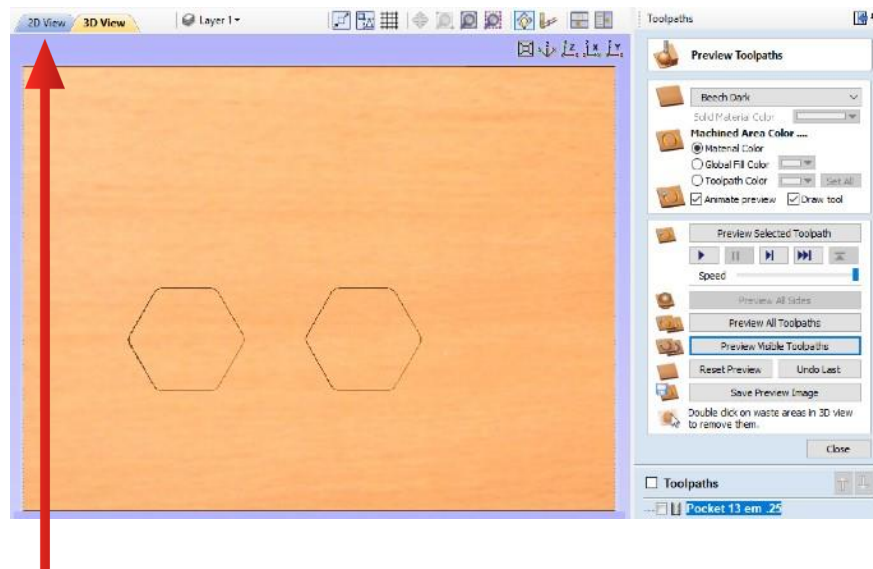
This will open the **Preview Toolpaths** panel and the **3D View** (simulator) window.



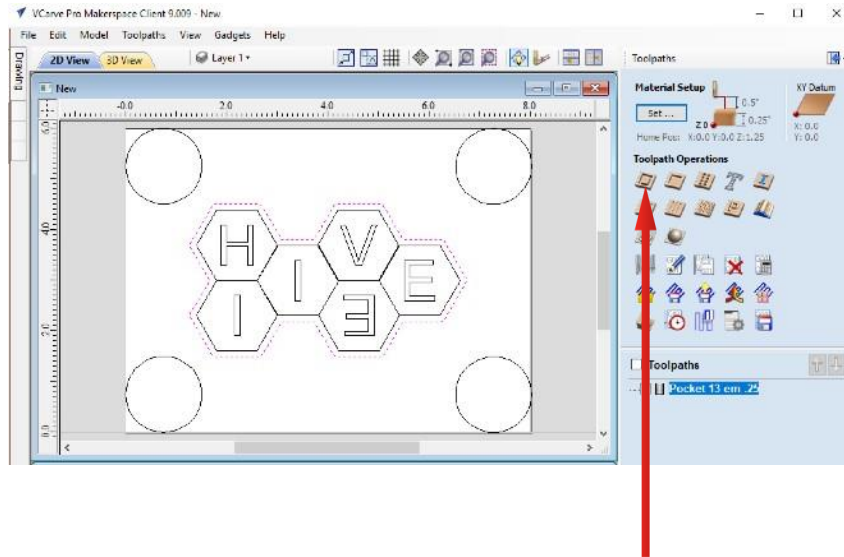
Be sure there is a check in the box in front of the tool path you want to simulate.
Click **Preview Visible Toolpaths**.

VCarve will show the results of the simulation of your toolpath, in the **3D View** window.

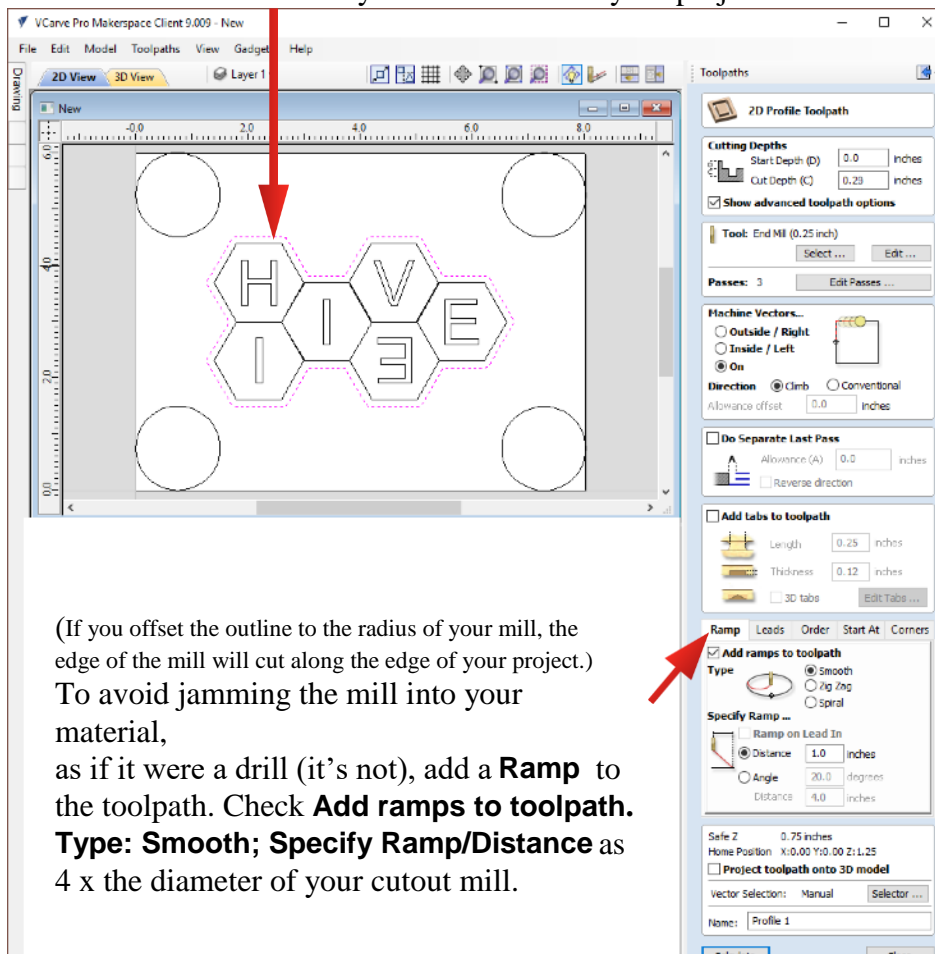
To set up the next toolpath, click **Close**.
You will go back to the **Toolpaths** panel.



To re-open the vector display, click on 2D View, at the upper left.



If you are cutting your project out of the blank, highlight the **Profile** tool path and the **Offset** vector you used to outline your project above.



(If you offset the outline to the radius of your mill, the edge of the mill will cut along the edge of your project.) To avoid jamming the mill into your material, as if it were a drill (it's not), add a **Ramp** to the toolpath. Check **Add ramps to toolpath**. **Type: Smooth; Specify Ramp/Distance** as 4 x the diameter of your cutout mill.

Set the **Start Depth** to 0.
Set the **Cut Depth** to material thickness + 0.02" to 0.04"
(Type "z=" then "0.04+")
Select a tool.
Click **OK**.

Edit the tool.

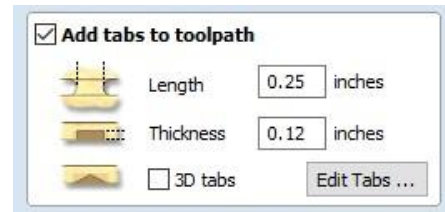
Under **Machine Vectors**, select **On**.

Finally, to prevent the project from flying off the table as it is cutout, add tabs to hold it in place.

In the sub-panel above **Ramps**, check **Add tabs to toolpath**.

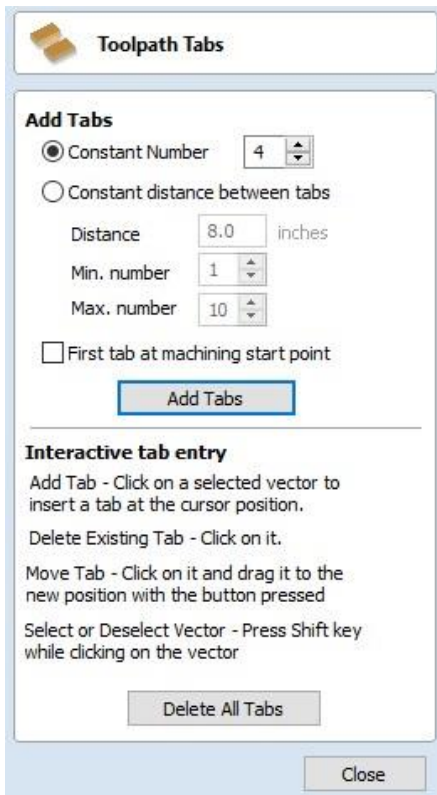
A good place to start is a **Length** of **0.25"** and a **Thickness** of **0.12 - 0.25"**

Click **Edit Tabs...**



This will open the **Toolpath Tabs** panel.

You should put some along each edge, about every 6"-12.



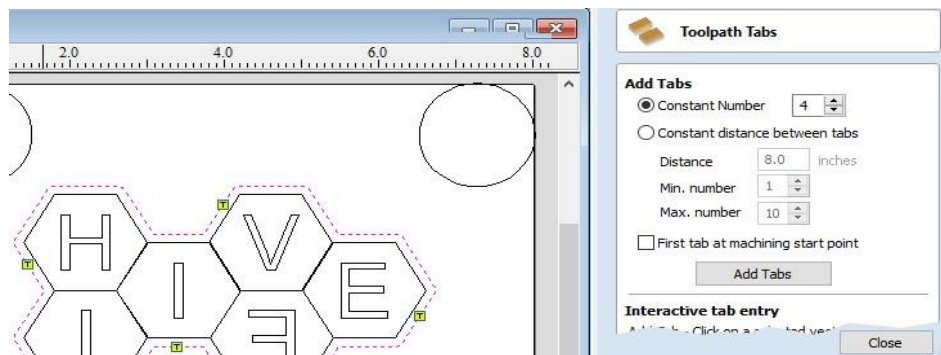
Using the “+” part of the cursor, click the vector to which you wish to add the tabs. A tab will appear where you click.

Small yellow squares will appear along the highlighted toolpath.

You can move the tabs with the cursor, or click on them to erase tabs.

Click **Close**.

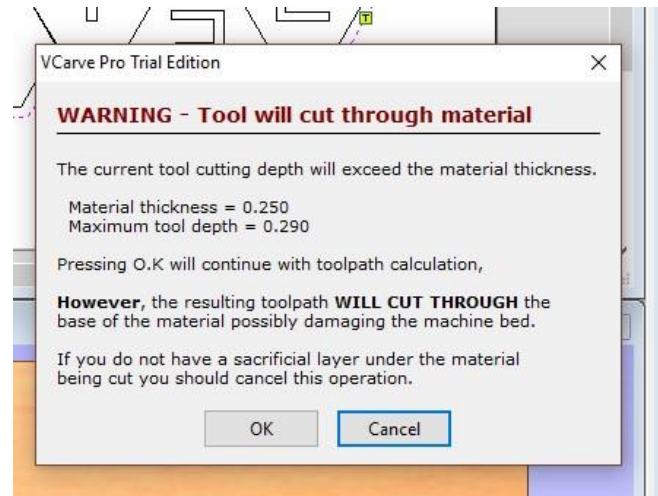
Click **Calculate**.



Be sure you get a **WARNING** from VCarve that you are going to cut through the material.

If you do not see this warning, your project will not be cut from the surrounding material.

Check that the **Maximum tool depth** is no more than **0.04"** (1mm) greater than the **Material thickness**.

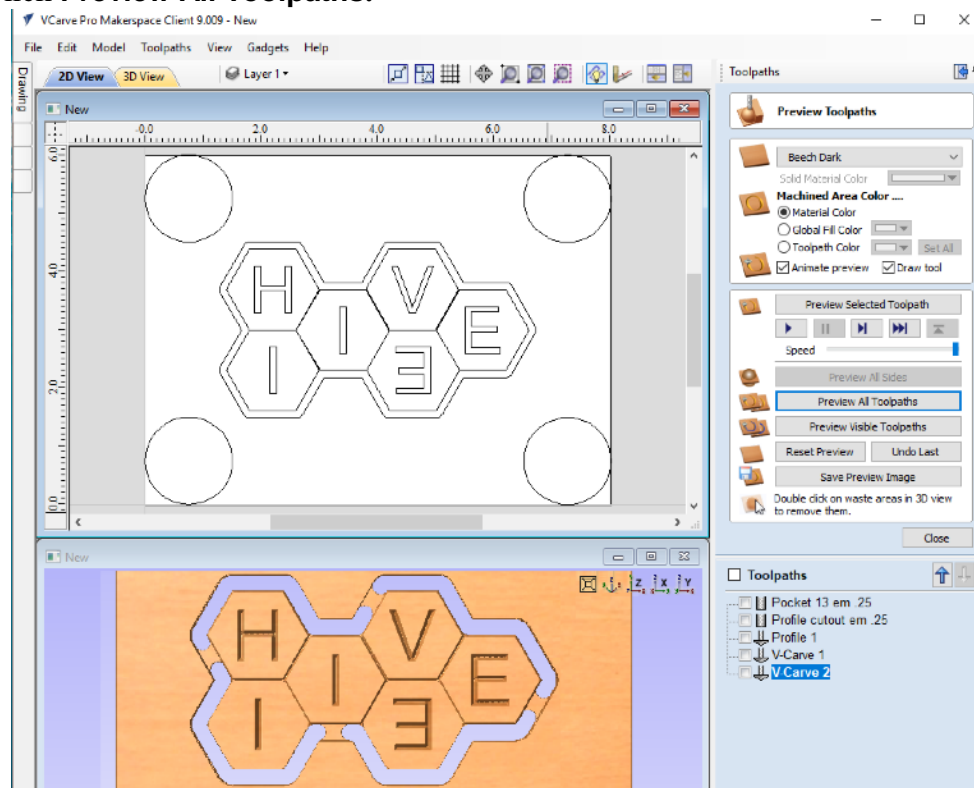


Click **OK** for the warning. The toolpath will be calculated.

This will open the **Preview Toolpaths** panel and the **3D View** (simulator) window.

To see the results of all your toolpaths at once,

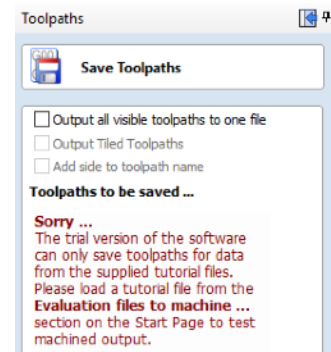
Click **Preview All Toolpaths**.



After all tool paths are calculated & checked, **Save** the file via the **File** menu. VCarve will save the file in the .crv format.

If you try to **Save Toolpaths** now, the software will refuse.

Take the .crv file you saved (e.g., to a USB drive) to Hive 1 3 and upload it to one of the computers running a registered copy of **VCarve Pro 9**.



Postprocessing

At Hive 1 3, launch VCarve Pro 9.

Open an Existing File to import your file into VCarve Pro 9.

When asked, affirm that you want to register (or re-register) it.

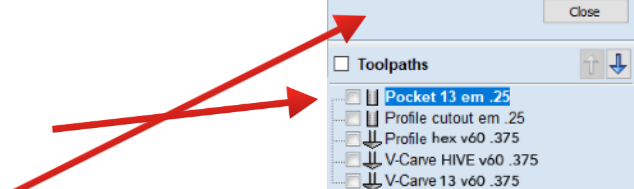
Go to the **CAM** window (**Toolpaths** panel), as above.

Click the floppy disk^(citation needed) icon to open the **Save Toolpaths** panel.

Be sure that **Mach2/3 Arcs (inch) (*.txt)** is the **Post Processor** being used.

Highlight each toolpath name in turn.

Click **Save Toolpath(s) to File**



for each toolpath.

You can save them to a USB drive or put them in a folder in the Z: Drive, DUMPSTOR

These are the GCode text files that you will load into **Mach3** to machine your project.